1 Introduction

Turbocharging technology has been widely applied to improve the performance of internal combustion engines (ICEs). In the past few decades, high-pressure-ratio turbocharging technology has played a vital role in turbocharged ICEs due to its advantages for increasing power density, lowering fuel consumption, and reducing emissions [1,2]. To satisfy rigid emission regulations, many technologies have been introduced to ICEs, and many of them need to be compatible with the high-pressure-ratio turbocharging technology. High rates of exhaust gas recirculation (EGR) and variable EGR technology have made an impact on controlling NOx emissions, but when a higher rate of EGR is used, a higher pressure ratio as well as a wider stable operational range of the turbocharger is needed to avoid an increase in fuel consumption and a decrease in power output [3,4]. Downsizing has been a useful technology used in ICE to improve CO2 emissions and fuel economy. The high-pressure-ratio turbocharger has an effective method for significantly downsizing ICEs [5]. When ICEs are operated in high-altitude areas, the high-pressure-ratio turbocharger is an indispensable component to help recover the engine power [6]. However, as the pressure ratio increases, the compressor (a core component of the turbocharger) suffers severe problems. For instance, the relative Mach number at the leading edges of the compressor blades increases, the flow inside the compressor becomes transonic, and compressors often suffer narrower flow margins because of the decrease in stall incidence and complex flow field [7], so the turbocharger no longer guarantees a sufficient stable flow range (SFR) to meet the ICEs requirements. Therefore, extending the compressor’s SFR is a critical issue for the development of high-pressure-ratio turbochargers.

The volute, which is an essential part of turbocharger centrifugal compressors, has a significant influence on the compressor performance. It is a nonaxisymmetric component that functions as a gas collector. At the design point, the volute usually shapes a uniform static pressure distribution along the circumferential direction, but it acts more like a diffuser at a lower mass flow rate or a nozzle at a higher mass flow rate; hence, the static pressure distribution along the volute passage is no longer uniform at off-design points. Much attention has been paid to the volute, and many studies have been carried out to investigate the volute flow phenomena and the interaction among the volute, diffuser, and impeller [8–13]. It has already been proved that the volute can influence compressor performance remarkably because its nonaxisymmetric geometry can make the flow fields in both the diffuser and the impeller significantly asymmetric [13].

To improve the performance of turbocharger centrifugal compressors, many studies have been done to help understand the flow mechanism inside the compressor and improve the compressor performance. Zheng and coworkers [14,15] investigated a high-pressure-ratio centrifugal compressor in detail. They found that an asymmetric flow field in the compressor decreased the SFR and that a higher rotating speed led to a narrower SFR. They also noticed a phase shift in the static pressure distortion between the diffuser and the impeller outlet and the splitter blade inlet. Shaaban and Seume [16] found that the volute had a significant impact on the flow field in the diffuser. They also detected the static pressure distortion

---

**Stability Improvement of a Turbocharger Centrifugal Compressor by a Nonaxisymmetric Vaned Diffuser**

The nonuniformity of the flow field induced by a nonaxisymmetric volute significantly degrades the stability of a turbocharger centrifugal compressor. In this paper, a nonaxisymmetric vaned diffuser is investigated as a nonaxisymmetric flow control method using both three-dimensional computational fluid dynamics (CFD) and experiment. The numerical study first focuses on the relationship between the flow field and the static pressure distortion, and the steady CFD results indicate that the positive static pressure gradient in the rotating direction facilitates flow separation in the vaned diffuser and induces a nonuniform flow field. A nonaxisymmetric flow control method with variable stagger and solidity of the vaned diffuser is developed to suppress the flow separation, and the guideline of the method suggests narrowing flow passages where the flow separates or closing diffuser vanes upstream of flow separations. Steady CFD also presents the flow field of the investigated turbocharger centrifugal compressor with volute, and flow separation is found in the flow passages near the volute tongue. Under the guidance of the nonaxisymmetric flow control method, several nonaxisymmetric vaned diffusers are designed to make the flow field uniform, which are believed to be beneficial for compressor stability. Finally, an experiment is carried out to validate the positive effects of the nonaxisymmetric vaned diffuser for stability improvement. The test data show that Non-AxisVD (with a nonaxisymmetric vaned diffuser) extends the stable flow range (SFR) of the compressor by 26% compared with the AxisVD (with an axisymmetric vaned diffuser), at the cost of acceptable decreases in the maximum total pressure ratio and peak efficiency. [DOI: 10.1115/1.4038875]

**Keywords:** compressor stability, nonaxisymmetric vaned diffuser, turbocharger, flow control method, CFD
induced by the volute at the diffuser outlet and recognized that this distortion could propagate to the impeller outlet; furthermore, they noticed that the distortion amplitude in the impeller outlet was larger than that in the diffuser outlet. Unsteady numerical methods were introduced into centrifugal compressor research by Fatsis et al. [17]. It could be concluded from their results that the static pressure distortion at the outlet influenced the aerodynamic parameters in the impeller inlet, such as inlet incidence. Zheng et al. [18] studied the propagation of volute-induced static pressure distortion and the amplitude effects on compressor performance, and their results showed that the distortion at the diffuser outlet could propagate to the upstream components and induce stall or flow separation in some flow passages, thereby influencing the SFR. Their study also indicated that the initial location of flow separation was related to the distortion amplitude. Zheng et al. [19] evaluated the influence of the volute on centrifugal compressor performance by using a numerical method. They found that the SFR at the design speed decreased by approximately 42% when applying the volute. The experimental method is another way to investigate the influence of a volute; the experimental results of Lin et al. [20] showed that a volute decreased the SFR and peak efficiency by 47% and 4.8%, respectively. To match the asymmetric flow field induced by the volute, some corresponding asymmetric flow control methods were developed to extend the centrifugal compressor’s SFR. Lin [21] studied a high-pressure-ratio compressor with a vaned diffuser and found the mechanism by which the asymmetric flow field affected stall and surge; he then established the design criteria for a vaned diffuser to extend flow stability. According to the design criteria, he redesigned an asymmetric vaned diffuser that extended the SFR by approximately 28.3% compared with a prototype. A nonaxisymmetric self-recirculation casing treatment (SRCT) was developed and investigated by Zheng et al. [14]. They noted that nonaxisymmetric SRCT had a larger potential for stability improvement than did axisymmetric SRCT, and they found that the surge flow rate of a compressor with nonaxisymmetric SRCT was approximately 10% lower than that of a compressor with axisymmetric SRCT.

For higher pressure ratios, a vaned diffuser is used in compressors to achieve higher efficiency. Despite the successes of nonaxisymmetric vanless diffusers and nonaxisymmetric SRCT in improving compressor stability, little information about the nonaxisymmetric vaned diffuser is available because the flow field in the vaned diffuser is more complex and the stability mechanism of the vaned diffuser is not fully understood. It was well known that the flow range of a centrifugal compressor with a vaned diffuser was much narrower than that with a vaned diffuser, so extending the flow range of compressors with vaned diffusers was inevitable and necessary. Considering the nonaxisymmetric flow field in a vaned diffuser and its noteworthy influence on compressor stability, we expected a significant improvement in compressor stability upon developing a nonaxisymmetric flow control method for a vaned diffuser. In this paper, the asymmetric flow field inside a vaned diffuser was investigated in detail, and a nonaxisymmetric flow control method for a vaned diffuser was developed. Based on the developed method, nonaxisymmetric vaned diffusers were designed and applied in a high-pressure-ratio turbocharger centrifugal compressor stage, and positive effects were detected in both numerical and experimental results.

2 Numerical Setup and Validation

2.1 Geometry and Mesh. The investigated centrifugal compressor is used in a turbocharger with a design speed of 90,000 rpm and a total pressure ratio over 3.0. Other detailed specifications of this compressor are shown in Table 1.

To discretize the fluid domain in the impeller and the diffuser, a multiblock structured grid is used. To guarantee a high quality of the mesh in areas around the impeller blade and the diffuser vane, O-blocks are used, whereas H-blocks are imposed in other areas.

Because the cross section of the flow passage in the volute is quasi-circular, butterfly blocks are applied to discretize the fluid domain of the volute. The finished mesh of the entire volute and a single impeller and diffuser passage are shown in Fig. 1. The numbers of mesh elements contained in the entire annulus of the impeller, the vaned diffuser, and the volute are approximately $7.3 \times 10^6$, $2.2 \times 10^6$, and $5.3 \times 10^6$, respectively. The distribution of the number of nodes in the impeller and diffuser meshes is displayed in Table 2. The volute mesh has 477 nodes in the tangential direction, and the number of nodes in the other two directions is 37. In addition, the entire mesh is of high quality: the minimum orthogonality is 16.41 deg, the maximum aspect ratio is less than 2000, and the maximum expansion ratio is less than 5.0. The validation of the mesh used in this investigation is discussed in Sec. 2.3.

2.2 Two Simulation Models and Settings. The ultimate purpose of this investigation is to improve the stability of a turbocharger centrifugal compressor by using a nonaxisymmetric vaned diffuser. To achieve this goal, the nonaxisymmetric flow control method is first established to make the flow field in a vaned diffuser uniform. This method is then used to design nonaxisymmetric vaned diffusers to extend the SFR of the compressor. During

<table>
<thead>
<tr>
<th>Parameters</th>
<th>Value (unit)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Impeller blade number</td>
<td>16</td>
</tr>
<tr>
<td>Inlet shroud diameter</td>
<td>62.70 (mm)</td>
</tr>
<tr>
<td>Impeller outlet diameter</td>
<td>100.00 (mm)</td>
</tr>
<tr>
<td>Impeller backsweep angle</td>
<td>34.50 (deg)</td>
</tr>
<tr>
<td>Diffuser vane number</td>
<td>16</td>
</tr>
<tr>
<td>Diffuser inlet diameter</td>
<td>117.60 (mm)</td>
</tr>
<tr>
<td>Diffuser outlet diameter</td>
<td>144.40 (mm)</td>
</tr>
<tr>
<td>Diffuser width</td>
<td>3.98 (mm)</td>
</tr>
<tr>
<td>Throat area of volute</td>
<td>3300 (mm²)</td>
</tr>
</tbody>
</table>

**Table 1 Main parameters of the compressor**

**Table 2 Distribution of mesh nodes in different directions**

<table>
<thead>
<tr>
<th>Directions</th>
<th>Impeller mesh</th>
<th>Vaned diffuser mesh</th>
</tr>
</thead>
<tbody>
<tr>
<td>Streamwise node number</td>
<td>141</td>
<td>41</td>
</tr>
<tr>
<td>Spanwise node number</td>
<td>53</td>
<td>37</td>
</tr>
<tr>
<td>Tangential node number</td>
<td>513</td>
<td>760</td>
</tr>
</tbody>
</table>

**Fig. 1 The finished mesh of the volute (top) and single passage mesh of the impeller and the vaned diffuser (bottom)**
the process of establishing the nonaxisymmetric flow control method, the “Distortion Model” is used to investigate the influences of some key parameters on the flow field. As shown in Fig. 2, the distortion model indicates that the simulation domain contains the impeller and the vaned diffuser only, whereas the simulation model with the volute domain is called the “Real Model.” These two simulation models use the same impeller mesh and diffuser mesh, and the difference between them is that the volute included in the real model is replaced by a sinusoidal static pressure distortion in the distortion model. Two advantages explain why the distortion model is appropriate for the investigation to establish the nonaxisymmetric flow control method: (1) a sinusoidal pressure distortion is imposed at the diffuser outlet to replace the complex volute-induced distortion, which is quite convenient for investigating the influence of diffuser design on the flow field and finding an effective flow control method to make the flow field uniform. (2) Since the volute is not included in the simulation domain, fewer grids of the mesh are calculated, and the computational costs and duration are significantly reduced.

For convenience when discussing the vaned diffuser, the flow passages and diffuser vanes are numbered from “1” to “16.” As shown in Fig. 3, a coordinate system is established with its origin
located at the rotational axis; its $x$-axis is antiparallel to the outlet pipe of the volute, and its $y$-axis is perpendicular to the volute outlet. The flow passage intersecting the $x$-axis is numbered passage 1 (red numeral), and the vane intersecting the $x$-axis is numbered vane 1 (blue numeral). Regions close to vane 5 and passage 5 will experience the maximum static pressure in the diffuser outlet in the distortion model, and passage 7 is close to the volute tongue in the real model.

Based on the three-dimensional steady compressible finite volume method, the numerical method used in this investigation solves the Reynolds-averaged Navier–Stokes equations with the commercial computational fluid dynamics (CFD) software NUMECA FINE/TURBO. A fourth-order Runge–Kutta scheme and a Central scheme are used for temporal and spatial discretization, respectively. The Spalart–Allmaras one-equation model is chosen for turbulence closure, and a multigrid procedure is applied to accelerate convergence.

The boundary conditions are set as follows: All of the solid walls in the impeller, diffuser, and volute are set to be nonslip and adiabatic. The total pressure, total temperature, and velocity directions are imposed as inlet boundary conditions. Setting the outlet conditions becomes slightly complex because of the difference between the distortion model and the real model. In the distortion model, a sinusoidal static pressure distribution along the circumferential direction is imposed at the outlet of the vaned diffuser. The pressure distribution can be expressed by the following equation:

$$
\begin{align*}
\rho(\theta) &= p_{ave} \cdot (1 + A \cdot \sin \theta) \\
A &= \frac{p_{max} - p_{min}}{2 \cdot p_{ave}} \times 100\%
\end{align*}
$$

where $\theta$ is the circumferential angle, $p_{ave}$ is the average static pressure, and $A$ is the amplitude of the static pressure distortion. As the influence of distortion amplitude is not considered in this investigation, it is set to a constant value ($A = 5\%$) for all of the operational conditions. The average static pressure $p_{ave}$ is changed to alter the operating point of the compressor. In the real model, the outlet condition settings are much simpler: constant static pressure is imposed at the volute outlet, and the operation point is changed by varying the imposed static pressure.

It would be better to perform an unsteady calculation method to investigate the flow in the distortion model and real model. However, unsteady computations require plenty of time and impose high computational costs, particularly when trying to find a new design method through a trial-and-error process. Therefore, a frozen rotor model is selected in this investigation to solve for the transition between the rotating impeller and the stationary vaned diffuser for both the distortion model and the real model.

### Validation

Before the numerical method can be used in the investigation, it is necessary to validate its ability to capture the real physical phenomena of the flow inside the compressor with the frozen rotor model. The mesh should be suitable for the turbulence model, and $y+$, which represents the nondimensional wall distance along the wall, should be close to 1.0 to apply the Spalart–Allmaras turbulence model. Figure 4 shows the $y+$ distribution, indicating that $y+$ is close to 1.0 at most of the solid walls, especially the walls of the impeller and the diffuser. The mesh used in this study is, therefore, quite suitable for the Spalart–Allmaras turbulence model.

It is well known that steady simulation is incapable of capturing the unstable boundary accurately on the surge side of the performance map, and it is not reliable for judging the surge points. In this investigation, the numerical results are compared with experimental results, and Table 3 presents the comparison at design speed of the maximum total pressure ratio, choke mass flow rate, peak efficiency, and SFR.

The definition of SFR is as follows:

$${\text{SFR}} = \frac{{m_{\text{choke}} - m_{\text{surge}}}}{{m_{\text{choke}}}} \times 100\% \quad (2)$$

where $m_{\text{choke}}$ and $m_{\text{surge}}$ are the mass flow rates at the choke condition and surge condition, respectively.

According to Table 3, good agreement is achieved between the experimental and numerical results for the maximum total pressure ratio, the choke mass flow rate, and the peak efficiency, and the relative differences in these three parameters are quite small (0.39%, 0.80%, and 1.39%, respectively). However, the relative difference in the SFR is higher than 70%. This disappointing result of the SFR comparison indicates the inability of steady CFD to predict the surge point, and the numerical SFR is incapable of functioning as an indicator of compressor stability. Therefore, another, more reasonable, indicator, i.e., the uniformity of the flow field, is selected to reflect the compressor stability in this investigation. As Zheng et al. [19] conclude, the nonuniform flow field induced by the nonaxisymmetric distortion deteriorates the stability of the compressor significantly. This study focuses on

### Table 3 Comparison between experiment and CFD at design speed

<table>
<thead>
<tr>
<th>Parameters</th>
<th>Experiment</th>
<th>CFD</th>
<th>Relative difference</th>
</tr>
</thead>
<tbody>
<tr>
<td>Maximum total pressure ratio</td>
<td>3.1412</td>
<td>3.1534</td>
<td>0.39%</td>
</tr>
<tr>
<td>Choke mass flow rate (kg/s)</td>
<td>0.3520</td>
<td>0.3548</td>
<td>0.80%</td>
</tr>
<tr>
<td>Peak efficiency (%)</td>
<td>72.0</td>
<td>73.0</td>
<td>1.39%</td>
</tr>
<tr>
<td>SFR (%)</td>
<td>37.9</td>
<td>11.3</td>
<td>70.18%</td>
</tr>
</tbody>
</table>

Fig. 4 The $y+$ distribution on all of the solid walls
reducing the circumferential nonuniformity in the vaned diffuser by controlling the main parameters, e.g., solidity and stagger angle. All of the work carried out in this investigation focuses on making the flow field uniform, and we assume that uniformity of the flow field benefits the stability of the compressor.

To judge whether the nonuniformity of the flow field in a vaned diffuser is reduced, the CFD simulation model with the rotor/stator interface to be set as *Frozen rotor* must be capable of predicting the structure of the flow field well. Only when the flow field structure is predicted accurately, it is possible to determine the uniformity of the flow field. Therefore, a validation of the CFD method is needed not only at the level of overall performance (e.g., efficiency and total pressure ratio) but also at the level of the detailed flow field. In this paper, the overall and detailed experimental results of the compressor case “Radiver” are used to validate the ability of the CFD method used in this investigation to capture the flow field details. Radiver is a centrifugal compressor with a wedge type diffuser developed by the Institute of Jet Propulsion and Turbomachinery of RWTH Aachen, and abundant flow field details inside this compressor have been obtained by advanced visual experiments. More details about the compressor Radiver and related experimental information are available in their published literature [22–25].

The settings of the numerical model used for the validation case Radiver are identical to the settings used for the real model. The number of passages in the case Radiver in the simulation model is different from that of the experimental Radiver, as the periods of the impeller and the diffuser are set to be the same to meet the requirements for applying the frozen rotor. However, the change in period does not change the flow structure (such as the wake-jet structure) greatly, so it is still capable of validation.

The flow fields of both provided numerical and experimental results are compared at two selected measurement locations: 2M′ and 4M. As shown in Fig. 5, the measurement plane 2M′ is located near the outlet of the impeller and intersects with the trailing edge of the impeller blade, whereas the measurement plane 4M is located at the inlet of the wedge diffuser. To make the comparison between CFD and experiment more scientific, the operating point where the mass flow rate is approximately 83% of the choke mass flow rate is selected as the comparison point, and the results of the velocity comparison at 80% rotational speed are shown in Figs. 6 and 7.

In Fig. 6, by comparing the relative velocities at the 2M′ measurement plane, it can be seen that the distribution of the magnitude of the relative velocities is quite similar between the numerical and experimental results. The main flow structures are captured by CFD: an area of low relative velocity located on the shroud side, although this area is slightly larger in the numerical result than in the experimental result, and high relative velocity areas are at the corners between the hub and the impeller blade in both the numerical and experimental results. Therefore, it can be concluded that the numerical method captures the main characteristics of the flow structure at 2M′.

![Fig. 5 Schematic of measurement locations in the compressor “Radiver” [22,25]](image)

![Fig. 6 Comparison of relative velocity at 2M′ at the operating point where $m/m_{choke} = 83\%$](image)
When comparing the absolute velocities at the inlet of the wedge diffuser (4M), one can see from Fig. 7 that the numerical prediction of the flow field structure is again quite good. Both the numerical and experimental results show low absolute velocity areas existing near the leading edge of the wedge type diffuser (marked by vertical dashed lines) and high absolute velocity areas in the region between two adjacent diffuser vanes. Small differences between CFD and experiment are evident; the low velocity areas predicted by CFD are larger than those measured by experiment, and the other is that the experimental high velocity area is closer to the upstream diffuser vane. Nevertheless, the flow structure is quite similar, indicating that the CFD can capture the main characteristics of the flow field.

The main function of the simulation model with the frozen rotor used in this investigation is not to accurately predict the surge point of the compressor but to predict details of the flow field structure. Based on the flow field structure, a nonaxisymmetric flow control method is developed, and the nonaxisymmetric vaned diffuser is designed to improve the stability of the compressor. The comparison between CFD and experiment for the compressor Radiver shows that the simulation method with frozen rotor is capable of predicting the flow field structure at the measurement locations 2M and 4M. Because 2M and 4M are located close to the interface between the rotating impeller and the stationary diffuser, the agreement between CFD and experiment in the flow field structures at these two locations indicates that the CFD model with frozen rotor can predict physical flow phenomena well both upstream and downstream of the interface. It can be concluded that the simulation model is reliable and suitable for the requirements of this investigation.

3 Results and Discussion

3.1 Nonaxisymmetric Flow Control Method With Distortion Model. To improve the stability of turbocharger centrifugal compressors with vaned diffusers by making the flow field uniform, it is necessary to develop a flow control method for vaned diffusers. In this section, two significant parameters, the solidity and the stagger, are investigated to study their influences on the uniformity of the flow field, and effective guidelines are established to help make the flow field uniform. Due to its simple distortion shape and low simulation cost, the distortion model is used in this section.

With the distortion model, the relationship between the flow separation and the static pressure distortion is investigated first. As shown in Fig. 8, with the sinusoidal static pressure distortion at the vaned diffuser outlet, the Mach number distribution in the vaned diffuser is nonaxisymmetric, and obvious differences appear between different flow passages. Taking the left half of the vaned diffuser as an example, passages 16, 1, 2, and 3 are full of low-momentum flow, which means that flow separation is possible. In contrast, no low-momentum flow appears in the other flow
passages. The circumferential locations of the flow passages with flow separation are between 330 deg and 60 deg, and the sinusoidal pressure distribution at the right side of Fig. 8 indicates that the static pressure has a large positive gradient in this region. Therefore, the static pressure in the downstream passage is higher than that in the upstream passage; thus, a pressure-difference force toward the upstream passage is generated (shown in the left side of Fig. 8). The flow is pushed toward the upstream passages by this force, so flow separation occurs on the pressure side (PS) in these passages. It is concluded that the positive static pressure gradient along the flow direction induces a pressure-difference force, which pushes the flow away from the PS, and facilitates the flow separation.

Another motivation to use the distortion model is to develop a flow control method to make the flow field uniform. An effective way to achieve this goal is to suppress the flow separation in certain passages. Pushing the flow back to the PS is recommended as a means to reduce the effects of the positive pressure gradient on flow separation. In this investigation, two important parameters, i.e., the solidity of the vaned diffuser and the stagger of the diffuser vane, are changed at different circumferential positions to push the separated flow back to the PS of the diffuser vanes. The diffuser solidity is changed by changing the central angle $\Delta \psi$ between two adjacent diffuser vanes (as shown in Fig. 9). Different schemes for changing the diffuser solidity have been designed, and their performance has been analyzed in detail with the distortion model. The numerical results show that when decreasing the central angle between separated passages, the flow separation can be suppressed and the flow field can be uniform. An example is given in Fig. 9; the central angles of passages 16, 1, 2, and 3 are decreased by 2 deg compared with the Origin case for the axisymmetric vaned diffuser, and the remaining passages share the circumferential spaces uniformly. When the Mach number distribution at the same mass flow rate is compared, it is obvious that flow separations no longer appear and that the uniformity of the flow field is improved remarkably. In the given example, the solidity of different passages is not completely the same, so the vaned diffuser is nonaxisymmetric. Based on changing the solidity of the vaned diffuser, the conclusion could be drawn that the uniformity of the flow field can be improved by narrowing the separated flow passages.

Controlling the flow field by changing the stagger of the diffuser vanes is similar to changing the solidity. In this investigation, the diffuser vanes are rotated around an axis located at the leading edge of the diffuser vane; in this way, the stagger of the diffuser vanes is changed without changing the position of the leading edge of the vane. As described earlier, many schemes with different vane stagger have been designed, and their performance was analyzed with the distortion model. The numerical results indicate that when a vane is closed to the adjacent downstream vane, the flow separation between these two vanes can be suppressed and the flow field can be uniform. Taking one of the designed schemes with a uniform flow field as an example, a comparison of the Mach number distribution with the Origin case at
the same mass flow rate is shown in Fig. 10. For the given example, positive numbers downstream of the diffuser vane mean that this vane is closed, whereas negative numbers mean that the vane is opened; for instance, $-2$ deg means that the vane is opened by two degrees. Figure 10 indicates that vanes upstream of the separated flow passages (passages 16−3) are all closed and that the flow separation disappears, thus obtaining a quite uniform flow field. In summary, closing the diffuser vanes upstream of the flow separation can make the flow field uniform.

In this section, two significant parameters, the diffuser solidity and the stagger of diffuser vanes, are changed at certain circumferential positions to address the nonaxisymmetric flow field, for example, flow separation at certain flow passages. Vaned diffusers with variable solidity and stagger distribution along the circumferential direction are designed and analyzed in detail. A nonaxisymmetric flow control method is summarized from numerical results and is effective for controlling flow separation and making the flow field uniform by changing the diffuser and the vane stagger. Narrowing the separated flow passages or closing the vane upstream of the flow separation can make the flow field uniform.

In this section, two significant parameters, the diffuser solidity and the stagger of diffuser vanes, are changed at certain circumferential positions to address the nonaxisymmetric flow field, for example, flow separation at certain flow passages. Vaned diffusers with variable solidity and stagger distribution along the circumferential direction are designed and analyzed in detail. A nonaxisymmetric flow control method is summarized from numerical results and is effective for controlling flow separation and making the flow field uniform by changing the diffuser and the vane stagger. Narrowing the separated flow passages or closing the vane upstream of the flow separation can make the flow field uniform.

The performance map of the AxisVD in terms of total pressure ratio and efficiency at design speed is shown in Fig. 11. The graph shows that the peak efficiency at design speed is 73.0%, the maximum total pressure ratio is 3.15, and the SFR is 11.3%. The comparison between the predicted performance of the real model and the experimental performance shown in Table 3 indicates that the peak efficiency and the maximum total pressure ratio are accurately predicted, whereas the prediction of SFR is not satisfying. Therefore, the real model focuses on changes in the flow field, whereas the change in SFR is investigated by experiment in this paper.

Because the volute-induced static pressure distortion has a significant effect on the uniformity of the flow field, the static pressure distribution along the circumferential direction of the vaned diffuser outlet is extracted from the CFD results. When the compressor operates near the surge point (marked as near-surge (NS) in the bottom graph in Fig. 11), the static pressure at the volute is not constant in the circumferential direction, as shown in Fig. 12. The numbers at the top of Fig. 12 are the flow passage numbers of the vaned diffuser, the curve with saw-toothed fluctuations shows the CFD results of the static pressure at each circumferential location, and the curve with two peaks shows the passage-average static pressure distribution. The passage-average static pressure is defined by the following equation:

$$p_{\text{ave}} = \frac{1}{\phi_2 - \phi_1} \int_{\phi_1}^{\phi_2} p \cdot d\theta = \frac{1}{\phi_2 - \phi_1} \sum_{i} p_i \cdot \Delta \phi_i$$

(3)

where $\phi_1$ and $\phi_2$ are the circumferential positions where the passage begins and ends, respectively. Letting $\phi_2 - \phi_1 = 360$ deg, the circumferential average static pressure over all passages is obtained (the horizontal dash line in Fig. 12).

The amplitude of the static pressure distortion indicates the degree of nonuniformity of the static pressure in the circumferential direction, and its definition is described in Eq. (1). According to the CFD result at the NS operating point, the amplitude of the static pressure distortion at the diffuser outlet is less than 2%. The static pressure distribution in Fig. 12 also shows that high static...
pressure appears at the outlets of passages 6 and 9, and this is caused by the geometric structure of the volute. As shown on the right side of Fig. 12, the volute tongue is near passage 7, and it interacts with the flow field. As a result, the static pressure in nearby areas increases.

The nonaxisymmetric static pressure distribution induced by the volute at the diffuser outlet also significantly affects the flow field. Figure 13 shows the Mach number distribution at 90% span in the vaned diffuser and the volute. When focusing on the low Mach number areas, low-momentum flow appears in passages 5, 6, and 8. Based on the areas circled with red dashes in Fig. 12, the static pressure presents a positive gradient at the outlet of these three flow passages. Therefore, it is obvious that the positive gradient leads to flow separation and makes the flow field nonaxisymmetric. Shown on the right side of Fig. 13 are the velocity vectors in the inlet of passage 8, which indicate that the flow shifts from the PS to the adjacent suction side. At the same time, backflow occurs in the leading-edge region of the diffuser vane, which is harmful to the compressor stability. If the flow separation could be suppressed and the flow field made uniform, it is highly possible that the compressor stability would be improved.

3.3 Design and Analysis of Nonaxisymmetric Vaned Diffusers. To improve the stability of a turbocharger centrifugal compressor with a vaned diffuser and volute, the flow field needs to be uniform and the flow separation in certain flow passages should be suppressed. With the distortion model, a nonaxisymmetric flow control method with variable diffuser solidity and variable stagger of diffuser vanes was developed that will be used as the design criterion to develop nonaxisymmetric vaned diffusers. As the flow separates in passages 5, 6, and 8, the guideline of the nonaxisymmetric flow control method suggests that these flow passages should be narrowed or that the diffuser vanes upstream of the flow separation should be closed. Many design schemes for nonaxisymmetric diffusers have been developed. However, the numerical analysis shows that changes made only in areas near the flow separation are insufficient to completely control the flow separation, so other designs are proposed with changes in more flow passages and diffuser vanes. Three designs are selected as examples to be displayed in this paper and their geometric information is provided in Fig. 14. An explanation of the table in Fig. 14 is needed to help understand the design of these nonaxisymmetric diffusers. Taking Non-AxisVD1 as an example, the

<table>
<thead>
<tr>
<th>Vane Number &amp; Passage Number</th>
<th>1</th>
<th>2</th>
<th>3</th>
<th>4</th>
<th>5</th>
<th>6</th>
<th>7</th>
<th>8</th>
<th>9</th>
<th>10</th>
<th>11</th>
<th>12</th>
<th>13</th>
<th>14</th>
<th>15</th>
<th>16</th>
</tr>
</thead>
<tbody>
<tr>
<td>Non-AxisVD</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Stagger</td>
<td>C1</td>
<td>C2</td>
<td>C2</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Solvity</td>
<td>N0.5</td>
<td>N0.5</td>
<td>N1.5</td>
<td>N0.5</td>
<td>E0.5</td>
<td>E1.5</td>
<td>E0.5</td>
<td>E0.5</td>
<td>N1</td>
<td>E1</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Non-AxisVD1</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Stagger</td>
<td>C1</td>
<td>C1</td>
<td>C1</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Solvity</td>
<td>N1</td>
<td>N1</td>
<td>N1</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Non-AxisVD2</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Stagger</td>
<td>O0.5</td>
<td>O0.5</td>
<td>O0.5</td>
<td>C1</td>
<td>C2</td>
<td>C3</td>
<td>O0.5</td>
<td>O0.5</td>
<td>O0.5</td>
<td>O0.5</td>
<td>O0.5</td>
<td>O0.5</td>
<td>O0.5</td>
<td>O0.5</td>
<td>O0.5</td>
<td>O0.5</td>
</tr>
<tr>
<td>Solvity</td>
<td>N1.5</td>
<td>N1</td>
<td>E0.5</td>
<td>E1.5</td>
<td>E1.5</td>
<td>E0.5</td>
<td>E0.5</td>
<td>N0.5</td>
<td>E1</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

Comment: “O1” → vane open 1’
“C1” → vane close 1’
“E1” → passage extend 1’
“N1” → passage narrow 1’

Schematic of Non-AxisVD1
entry “C1” in the “stagger” row, column “5”, means that vane 5 is closed by 1 degree. Similarly, “N1” in the “solidity” row, column “5”, means that passage 5 is narrowed by decreasing the central angle by 1 deg.

The effects of the selected designs on the flow field are analyzed numerically. Figure 15 shows the Mach number distribution inside both the vaned diffuser and the volute at 90% span. For all of the presented cases, including the AxisVD, the Mach number contour is extracted at the same mass flow rate (90% of the choke mass flow rate). Compared with the AxisVD, flow separations in passages 5, 6, and 8 are successfully suppressed with the nonaxisymmetric vaned diffuser in Non-AxisVD and Non-AxisVD1, and their flow fields are quite uniform. Nevertheless, despite the successful suppression of flow separation in passage 8, the flow field of Non-AxisVD2 is not better than that of the AxisVD, as more flow separation exists in flow passages upstream of the volute tongue, and the flow field becomes less uniform, which is thought to be harmful to the compressor stability. Although Non-AxisVD and Non-AxisVD1 both obtain better flow fields than does the AxisVD, Non-AxisVD is superior to Non-AxisVD1, as the flow field of Non-AxisVD1 shows small flow separations appearing at the trailing edge of some diffuser vanes. Thus, Non-AxisVD is considered the most successful design with nonaxisymmetric stagger and solidity distribution in the circumferential direction, and its effects on stability improvement will be validated by experiment.

Before the experimental validation, more analysis of the flow field was carried out. Figure 16 shows the solidity and stagger of Non-AxisVD at different circumferential positions. The curves indicate that passages 1, 2, 3, 4, and 15 have been narrowed, as the central angle is less than 22.5 deg; however, vanes 4, 5, and 6 were closed because the vane stagger angle difference ($\alpha - \alpha_{axis}$, where $\alpha_{axis}$ is the stagger of the axisymmetric vaned diffuser) is positive. Narrowing the flow passage and closing upstream vanes are effective methods for pushing the separated flow back to the PS of the diffuser vane and suppressing the flow separation, and these are the reasons why the flow separations in passages 5 and 6 disappear. Nevertheless, the curves in Fig. 16 show that neither the solidity nor the stagger in passage 8 is changed, but the flow separation in passage 8 also disappears. The suppression of the flow separation in passage 8 is caused not by the change of solidity or stagger but by the redistribution of the mass flow rate through the flow passage. As shown in Fig. 17, because passages 5, 6, and 7 are narrower in Non-AxisVD, the mass flow rate through these passages is smaller than in the AxisVD; as a result, more flow passes through the downstream flow passages (like passage 8). To allow more flow to pass through, the flow must go faster and therefore the Mach number also changes. As shown in the bottom of Fig. 17, the absolute Mach number at the inlet of passage 8 in Non-AxisVD increases to become much higher than that of the AxisVD, which means the flow through passage 8 is more energetic and is less likely to separate. At the same time, the incoming flow condition at the leading edge of the diffuser vane, which greatly affects the flow stability, is also changed. The velocity vectors of the AxisVD and Non-AxisVD near the leading edge of vane 8 are presented in Fig. 18. Obviously, compared

![Figure 15 Comparison of Mach number distributions between the AxisVD and the selected non-axisymmetric design schemes (90% span)](image_url)

![Figure 16 Solidity and stagger distribution at different circumferential positions of Non-AxisVD](image_url)
with the AxisVD, the velocity vectors in Non-AxisVD are closer to the radial direction, and no backflow appears at the leading edge of vane 8, as the flow no longer travels from PS to suction side. It is believed that the flow in the vaned diffuser of Non-AxisVD is more stable and that the compressor stability will benefit greatly from this flow condition.

4 Experimental Comparisons

Based on the assumption that compressor stability is related to the degree of flow field uniformity, some nonaxisymmetric vaned diffusers are designed using the developed nonaxisymmetric flow control method. According to the results of flow field comparison between the AxisVD and the newly designed schemes, Non-AxisVD obtains the best uniform flow field inside the vaned diffuser and the volute, indicating that the compressor with this vaned diffuser design should provide superior performance in the SFR. However, no direct evidence proves the correctness of the relationship between stability and uniformity of the flow field, and it is doubtful whether Non-AxisVD indeed extends the SFR of the compressor stage. Therefore, experimental validation is necessary.

4.1 Experimental Setup. A schematic of the test rig facility is shown in Fig. 19. The entire system contains the centrifugal compressor as well as upstream and downstream pipelines. The geometric characteristics of the tested compressors are identical to those simulated by the real model, and detailed information is presented in Table 1. Pictures of the impeller, the diffuser, and the volute are displayed on the right side of Fig. 19. The diffuser is bolted to the volute casing and can easily be removed from the volute, so it is convenient to install different diffusers for different experimental cases without replacing the impeller and the volute. During the experiment, the turbine extracts energy from the high pressure and high temperature gas heated in a combustion chamber in front of the turbine and drives the compressor at the required rotating speeds. The outlet valve opening is adjusted during the experiment to change the operating conditions, and the compressor performance map is obtained from flow parameters measured at the inlet and outlet of the compressor. As shown in Fig. 19, two measurement sections denoted S1 and S2 are selected at the inlet and outlet pipelines, respectively. The mass flow rate, the total/static pressure, and the total temperature are measured at S1 and S2 with appropriate probes. The rotating speed, the ambient pressure, and the ambient temperature are also obtained. A lemniscate flowmeter with measurement error less than 1.0% is installed at S1 and a vortex shedding flowmeter with measurement error less than 1.5% is installed at S2 to measure the mass flow rate at the compressor inlet and outlet, respectively. The measurement error of the temperature probes installed at sections S1 and S2 is $0.25 \text{ K}$, while the measurement error of the pressure probes is $0.05\%$ of the measurement range ($-30$–$30 \text{kPa}$ at section S1 and $0$–$700 \text{kPa}$ at section S2).

4.2 Performance Comparison at Design Speed. The experiment is carried out to validate the expected positive effects of the nonaxisymmetric vane diffuser on the compressor stability. The numerical results show that the design scheme of Non-AxisVD
related to the degree of flow field uniformity is reasonable. Therefore, the nonaxisymmetric vaned diffuser improves the stability of the compressor, and the assumption that stability is extended the SFR of the compressor by 26% at design speed.

Non-AxisVD and the AxisVD are 37.50% and 47.26%, respectively. Compared to the AxisVD, the experimental SFRs of the AxisVD and the Non-AxisVD obtain the best flow field among all of the selected nonaxisymmetric vaned diffuser designs, and it is believed to have the potential to improve the stability of the centrifugal compressor. Therefore, the AxisVD and Non-AxisVD are tested during the experiment, and their performance at design speed is compared in detail to assess the improvement in stability. The experimental performance map of total pressure ratio and efficiency at design speed is shown in Fig. 20. The SFR of Non-AxisVD is wider than that of the AxisVD. The experimental SFRs of the AxisVD and Non-AxisVD are 37.50% and 47.26%, respectively. Compared with the axisymmetric vaned diffuser of the AxisVD, the newly designed nonaxisymmetric vaned diffuser of Non-AxisVD extends the SFR of the compressor by 26% at design speed. Therefore, the nonaxisymmetric vaned diffuser improves the stability of the compressor, and the assumption that stability is related to the degree of flow field uniformity is reasonable.

According to Fig. 20, the choke mass flow rate of Non-AxisVD is slightly smaller than that of the AxisVD. This change is mainly caused by closing some of the diffuser vanes, which decreases the area of the diffuser throat. The relative change in choke mass flow rate is approximately 0.5%, and it is so small that its influence on the match between impeller and diffuser could be negligible. Thus, it is evident that the improvement in stability is due not to the match change but to the nonaxisymmetric vaned diffuser.

Although the experimental result validates the stability improvement of the nonaxisymmetric vaned diffuser, it should be noted that both the total pressure ratio and the efficiency decrease slightly when the nonaxisymmetric vaned diffuser is applied (as shown in Fig. 20). The difference in maximum total pressure ratio (3.143 versus 3.104) is 1.2%, and the difference in peak efficiency (72.00% versus 71.05%) is 1.3%. Both of these differences are acceptably small when the improvement in stability is taken into account. As the total pressure ratio and the efficiency are not only related to the uniformity of the flow field but to many other aspects, the reason for the decrease in efficiency and pressure ratio needs further investigation. More work on nonaxisymmetric vaned diffuser would help design nonaxisymmetric vaned diffusers that could significantly improve the stability of the compressor without decreasing the total pressure ratio and efficiency.

5 Conclusions

A three-dimensional numerical investigation has been carried out on a turbocharger centrifugal compressor. By locally changing the stagger and solidity of the vaned diffuser, a nonaxisymmetric flow control method is established. According to the method, several nonaxisymmetric vaned diffusers are designed to extend the SFR of the compressor at design speed, and the positive effect on stability is validated by experiment. The main conclusions are as follows:

1. When the static pressure at the diffuser outlet varies in the circumferential direction, certain flow passages will suffer a positive pressure gradient in the rotating direction, thus a force induced by the pressure difference generates and pushes the flow away from the PS of the adjacent downstream diffuser vane, facilitating flow separation.
2. Flow separation in the vaned diffuser can be suppressed to make the nonaxisymmetric flow field uniform by locally changing the solidity and/or the stagger of diffuser vanes. The effective and recommended guideline to achieve this goal is to narrow the flow passages where the flow separates or to close the diffuser vane upstream of the flow separation.
3. For the investigated turbocharger centrifugal compressor with volute, high static pressure appears in areas near the volute tongue at the diffuser outlet. Flow separation is detected in flow passages upstream and downstream of the volute tongue, which is caused mainly by the volute-induced static pressure distortion as the static pressure presents a positive pressure gradient in these regions.
4. To control the flow separation and improve the stability of the compressor, the stagger and the solidity of the vaned diffuser are changed locally according to the guideline mentioned in conclusion (2); in this way, several nonaxisymmetric vaned diffusers are designed and their performance is investigated in detail. The numerical results show that Non-AxisVD successfully controls flow separation and obtains a uniform flow field, which improves the stability of the compressor significantly.
5. Experimental comparison between the AxisVD and the Non-AxisVD validates the positive effects of the nonaxisymmetric vaned diffuser on improving the stability of the compressor. Experimental results show that Non-AxisVD (SFR = 47.26%) extends the SFR by 26% compared with the AxisVD (SFR = 37.50%) with acceptable decreases in efficiency and total pressure ratio. The nonaxisymmetric vaned diffuser shows great potential in improving the stability of centrifugal compressors and more detailed investigations are needed on this topic.

Acknowledgment

This research was supported by the IHI Corporation, Yokohama, Japan. The authors of this paper would like to thank the Institute of Jet Propulsion and Turbomachinery of RWTH Aachen for providing the test case “Radiver”, whose experimental results are used to validate the CFD method in this investigation.

Funding Data

- National Natural Science Foundation of China (Grant No. 51176087).

Nomenclature

- $A$ = amplitude of static pressure distortion
- $c$ = absolute velocity
\[ m = \text{mass flow rate} \]
\[ Ma = \text{Mach number} \]
\[ N = \text{rotational speed} \]
\[ p = \text{static pressure} \]
\[ SFR = \text{stable flow range} \]
\[ w = \text{relative velocity} \]
\[ y^+ = \text{nondimensional wall distance} \]
\[ z = \text{stagger of diffuser vane} \]
\[ \Delta \psi = \text{central angle of a diffuser passage} \]
\[ \theta = \text{circumferential angle} \]
\[ \phi = \text{circumferential position} \]

**Subscripts**
- \( \text{ave} \): averaged value
- \( \text{max} \): maximum value
- \( \text{min} \): minimum value
- \( \text{surge} \): surge condition

**References**


---

\[ \text{Xu, C., and Amano, R. S., 2006, “Eliminating Static Pressure Distortion by a Large Cut Back Tongue Volute,” ASME Paper No. GT2006-90001.} \]